Review Paper:

A Comprehensive approach using CFD and GIS for dam break risk analysis: A case study on Nagarjuna Sagar earthen dam

Nanna Sri Ramya[1*], Reddy Madhusudhan M.[1], Reddy Suryaprakash V.[1], Saikumar R.[2] and Tanuja D.V.[3]
1. Department of Civil Engineering, Institute of Aeronautical Engineering, Dundigal, Hyderabad, Telangana, INDIA
2. Department of Arts, K L Deemed to be University, Andhra Pradesh, INDIA
3. Department of Civil Engineering, Malla Reddy Engineering College, Maisammaguda, Hyderabad, Telangana, INDIA
*n.sriramya@iare.ac.in

Abstract

This study presents a comprehensive approach utilizing Computational Fluid Dynamics (CFD) and Geographic Information Systems (GIS) to assess dam break risks with a specific focus on the Nagarjuna Sagar earthen dam. The study examines the numerical analysis of water flow dynamics resulting from a dam break. To depict this phenomenon, a two-dimensional numerical model employing the volume of fluid method was developed. The mathematical framework incorporates the Reynolds-averaged incompressible Navier-Stokes equations with the turbulent k-e model and was employed. The computational approach employed in this study is the SIMPLE algorithm. The CFD analysis involved meshing with 42,282 nodes and 41,680 elements, while boundary conditions included one velocity inlet and three pressure outlets open to the atmosphere. The flow was initialized with a 5 m dam site, scaled down at 1:10 (1 meter in the model representing 10 meters in reality). Data were collected for various time intervals (1s, 2s, 2.5s, 3s and 5s), The results of this study reveal the flow velocity patterns following dam break events for the specified time intervals.

The outcomes are presented in terms of velocity magnitude vectors, static pressure, dynamic pressure, total pressure, turbulence kinetic energy, volume fraction contours and velocity magnitude profiles at different distances from the dam site (1m, 2m, 5m and 7m). A total of 250 iterations were performed to achieve these results, providing valuable insights into the dynamics of dam break scenarios. This research contributes to a better understanding of the potential consequences of dam failures and aids in improving risk assessment and mitigation strategies for dams.

Keywords: Dam break, Computational fluid dynamics, Geographic information systems, Risk analysis, Nagarjuna Sagar dam, Turbulence k-e model, Volume fraction.

Introduction

Dams play a pivotal role in water resource management, providing essential benefits such as irrigation, hydroelectric power generation and water supply[10]. However, the potential risks associated with dam failures have garnered increasing attention in recent years. The consequences of a dam break can be catastrophic, resulting in loss of life, extensive property damage and severe environmental impacts[4]. Therefore, there is a pressing need for comprehensive dam break risk analysis to assess and mitigate these potential hazards. The Nagarjuna Sagar earthen dam, a significant hydraulic structure located in Nalgonda district of Telangana (figure 1), stands as a critical component of water resource management in the region. With the increasing demand for water resources and the changing climate patterns, the evaluation of dam safety and the understanding of potential dam break scenarios become imperative[40].

This study presents a comprehensive approach to address the complex issue of dam break risk analysis, leveraging the power of Computational Fluid Dynamics (CFD) and Geographic Information Systems (GIS). CFD allows for the simulation of fluid flow dynamics, enabling a detailed examination of how water behaves in and around the dam during a breach event. GIS. It also provides valuable spatial data and analysis tools for a holistic assessment of the geographical context in which the dam is situated[44].

In the pursuit of enhancing dam safety and preparedness, this research focuses on the Nagarjuna Sagar earthen dam as a case study. The study employs CFD techniques with a mesh configuration comprising of 42,282 nodes and 41,680 elements and boundary conditions incorporating velocity inlets and pressure outlets. Additionally, data is collected at various time intervals to understand the dynamics of the flow during and after a dam break. The motivation behind this research is to contribute to the field of dam safety and risk assessment by providing a detailed analysis of potential dam break scenarios[13,44].

By combining CFD and GIS, this study aims to deliver valuable insights into flow patterns, pressure variations and other critical parameters in the event of a dam failure. These findings can inform more effective risk mitigation strategies, emergency preparedness plans and infrastructure improvements, ultimately safeguarding lives and valuable resources in the region.

Significance of dam break risk analysis: The significance of dam break risk analysis is underscored by the potential

https://doi.org/10.25303/173da035047

catastrophic consequences that can arise from dam failures including loss of life, extensive property damage and severe environmental impacts. This analysis provides essential insights and information to assess and mitigate these risks effectively. In this context, our specific focus is on the Nagarjuna Sagar earthen dam, situated at a latitude of 16°34'23"N and longitude of 79°18'47"E, in a hydrologically sensitive region with the following key hydrological and structural characteristics:

Hydrological parameters:

- The catchment area at the dam site covers a vast expanse of 83,083 square miles (215,185 square kilometres), making it susceptible to diverse hydrological factors.
- The maximum annual rainfall in the catchment area reaches 35 inches (889 millimeters), highlighting the potential for high-volume inflow into the reservoir.
- The maximum observed flood recorded stands at a staggering 30,050 cubic meters per second (10.61 lakh C/s), underscoring the significance of assessing dam safety under extreme conditions.
- Even during dry weather, the minimum flow is substantial at 2.80 cubic meters per second (100 C/S).
- For comprehensive risk assessment, a 1000-year design flood of 58,340 cubic meters per second (20.60 lakh C/s) is considered, indicating the need to evaluate the dam's resilience under extreme, rare events.
- The routed flood at 45,310 cubic meters per second (16.00 lakh C/s) presents a challenging scenario for dam management and preparedness.

Details of Nagarjuna Sagar reservoir:

- The full reservoir level (F.R.L.) reaches +590.00 feet (+179.832 meters), emphasizing the considerable storage capacity of the dam.
- The maximum water level is +594.00 feet (+181.051 meters), highlighting the potential risk to downstream areas in case of a dam breach.
- The dead storage level at +400.00 feet (+121.920 meters) plays a critical role in water resource management and flood control.
- The maximum draw down level (M.D.D.L.) is defined at +510.00 feet (+155.450 meters), indicating the range of water levels that must be carefully managed.
- The expansive water spread area of 110 square miles (285 square kilometres) demonstrates the significant impact a dam breach could have on surrounding regions.

Details of earth dam:

- The earthen dam consists of a left and right earth dam with lengths of 8,400 feet (2560.32 meters) and 2,800 feet (853.44 meters) respectively.
- The maximum height of the dam stands at 85 feet (25.908 meters), underscoring its scale and potential consequences in case of structural failure.
- The top width of 30 feet 6 inches (9.296 meters) and a top level of +610.00 feet (+185.928 meters) provide essential structural details that influence the dam's hydraulic behaviour.

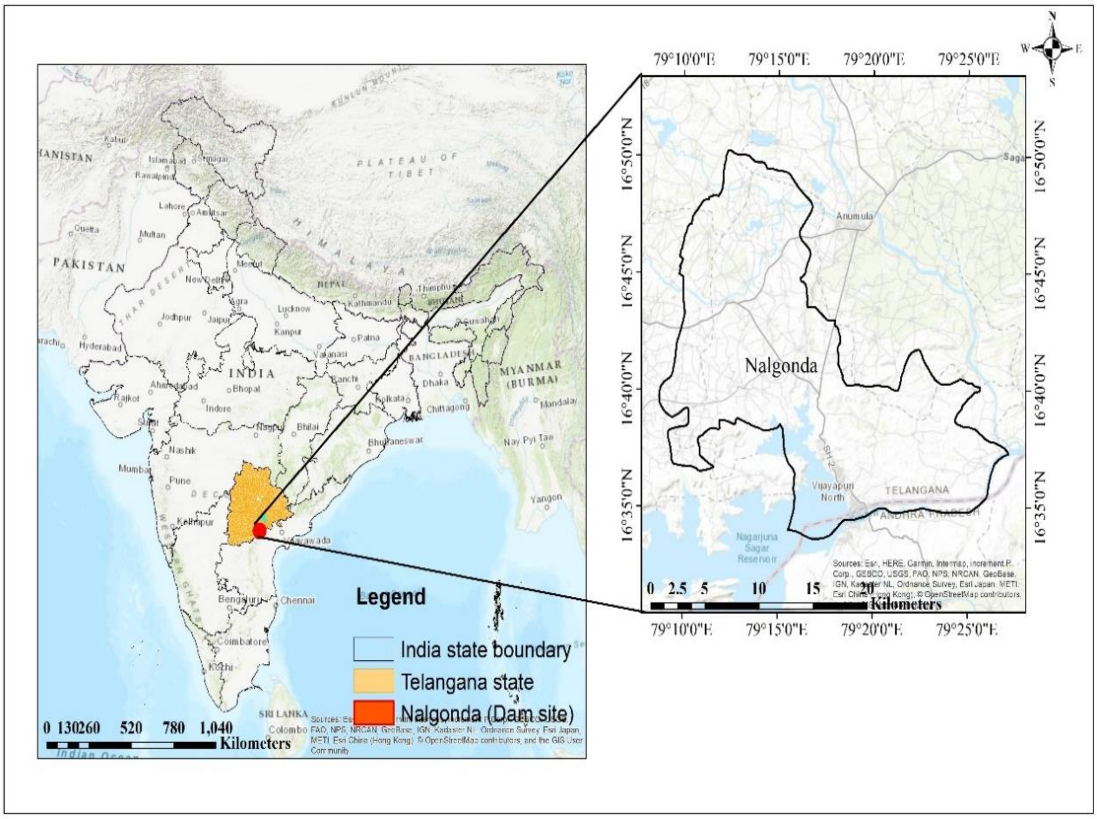


Figure 1: Geographical location map of the Nagarjuna sagar dam

Given these hydrological and structural characteristics, the Nagarjuna Sagar earthen dam presents a critical case study for dam break risk analysis. The research aims to provide valuable insights into the dam's behaviour under various scenarios, enhancing our understanding of its safety and ultimately contributing to improved risk assessment and mitigation strategies for this essential water resource infrastructure. Geographical location of the Nagarjuna sagar reservoir is shown in figure 2.

A critical appraisal on reviewed literature: The integration of CFD and GIS has emerged as a powerful tool in the realm of dam break risk analysis, with particular emphasis on its practical applications in safeguarding critical infrastructure such as the Nagarjuna Sagar earthen dam. This innovative approach represents a significant step forward in the field of dam safety assessment and management, offering the potential to enhance our understanding of complex hydraulic phenomena and their spatial distribution. As we delve into the body of literature addressing this comprehensive approach, it becomes imperative to critically appraise the existing research, evaluating the strengths and weaknesses of the methodologies employed, the quality of data utilized and the robustness of the findings.

This critical appraisal aims to provide insights into the current state of knowledge surrounding CFD-GIS integration for dam break risk analysis, shedding light on the key advancements, limitations and areas requiring further investigation in the context of the Nagarjuna Sagar earthen dam.

In 2014, Sun et al[36] emphasized the critical need for safety engineers and decision makers to conduct comprehensive risk analyses of dam-break floods, due to the significant human and property losses associated with such events. Previous flood routing simulations primarily employed one- or two-dimensional models, overlooking the complexities of three-dimensional terrain which substantially impact risk assessment accuracy. The authors propose an integrated approach to dam-break risk analysis in complex terrain which combines a three-dimensional turbulence mathematical model, an attribute measure approach for consequence assessment and a combined weight method with TOPSIS-based analysis for risk ranking. This comprehensive model is validated and applied to a case study involving five reservoirs in China's Haihe River Basin, providing a robust framework for informed decision-making in dam safety and reinforcement prioritization.

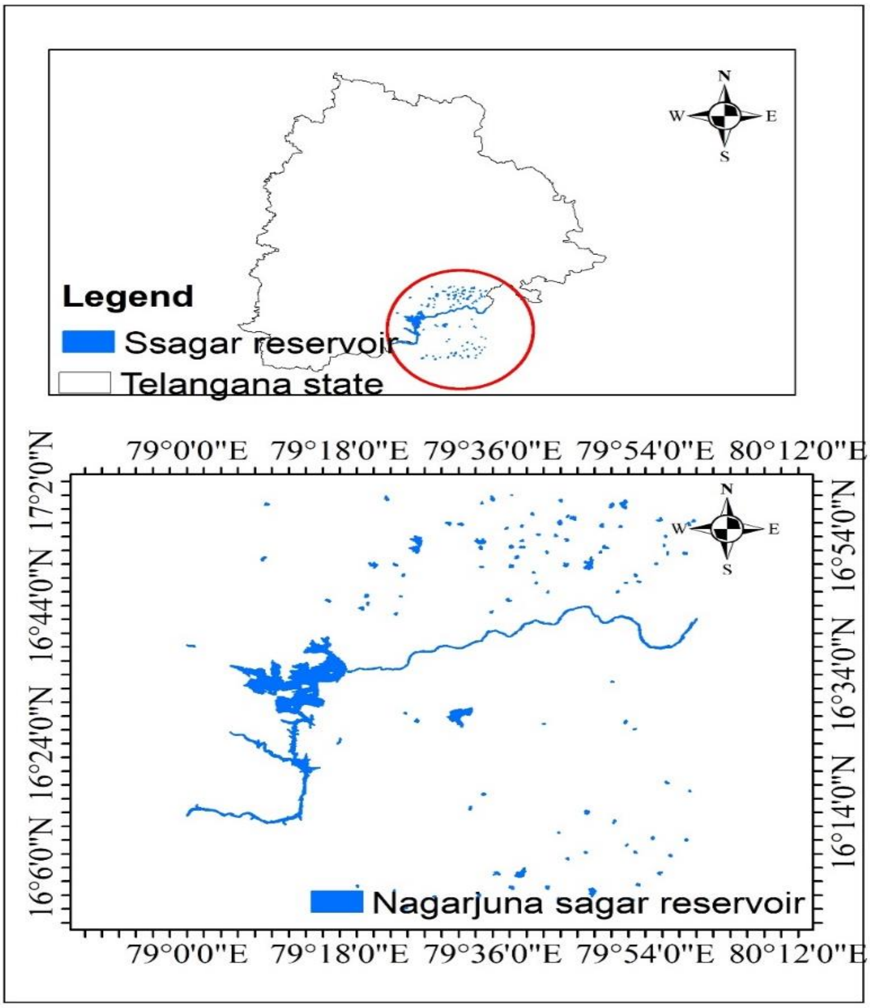


Figure 2: Nagarjuna sagar reservoir and upstream and downstream flow lines

In their 2014 study, Zhou et al[46] highlighted the critical need for assessing the risks associated with dam-break floods in the Haihe River Basin, China. They introduce a novel three-dimensional mathematical model that incorporates 3D geological data into CFD simulations to account for the complex terrain. Their findings emphasize the importance of timely warnings and public awareness in minimizing loss of life during dam breaks, with alarms being more effective than understanding alone. The study employs synthetic assessment and the TOPSIS method to rank the severity of consequences for various reservoirs, demonstrating that the Dongwushi reservoir poses the greatest risk. The authors affirm the stability and reliability of their evaluation model, while also acknowledging ongoing efforts to enhance CFD-based tools for simulating dam breaches in urban environments downstream.

In Kocaman et al[12] study, the sequential dam-break problem in a steeply inclined rectangular channel with consecutive twin reservoirs was examined both experimentally and numerically using image processing and the Smoothed Particle Hydrodynamics (SPH) method respectively. The results showed satisfactory agreement between the two approaches and revealed five distinct flow stages during the dam-break event. Additionally, the study explored the impact of camera quality on measurement accuracy, concluding that smartphone cameras can effectively replace professional cameras for such experiments. The research highlighted the importance of accurate camera positioning, calibration and lighting. Future investigations should consider varying parameters and the particulate nature of SPH, while the provided data serves as a valuable resource for further research in this field.

In Biscarini et al[6] study, the selection of a suitable model for dam break flood routing in hydraulic engineering is explored. Two models, SW and RANS-VOF, are tested on various dam break scenarios. The RANS-VOF model is found to better simulate transient free surface behavior, although it is computationally more intensive. When compared to experimental data, the three-dimensional RANS-VOF model consistently performs well, highlighting its potential for accurately representing complex flow phenomena, especially in situations where the simplified SW model may lead to significant errors. This study underscores the value of advanced computational resources for three-dimensional modeling in hydraulic engineering applications[35].

In summary, a coupled computational approach was used to analyze dam break scenarios involving mixtures of non-Newtonian liquids and granular particles[3]. The study emphasized the importance of considering solid-liquid interactions, demonstrating that a single-phase continuum approach is insufficient for describing the entire collapse process. The research revealed that the viscosity of non-Newtonian liquids and their density relative to particles significantly influence the flow dynamics[11]. Additionally, the study highlighted the impact of solid-liquid interactions on the movement of fronts between solid and liquid phases in mixtures and their effects on kinetic energy changes in particles. Ultimately, the findings indicated that accounting for solid-liquid interactions can alter the flow regimes observed in both liquid and granular systems in the context of dam break scenarios[5].

The primary objectives of this study are as follows:

1. Assessment of Dam Safety: To comprehensively assess the safety and structural integrity of the Nagarjuna Sagar earthen dam under various hydrological scenarios including extreme events like the 1000-year design flood.

2. Evaluation of Dam Break Risks: To quantify and analyse the potential risks associated with dam failure including the assessment of downstream flood inundation extents, flow velocities and the impact on populated areas.

3. Identification of Mitigation Measures: To identify potential structural and operational measures that can mitigate dam break risks and enhance the dam's resilience in the face of extreme hydrological events.

Governing Equations of Computational Fluid Dynamics (CFD): In the field of Computational Fluid Dynamics (CFD), the fundamental equations that govern fluid flow behaviour are the Navier-Stokes equations. These equations describe the conservation of mass and the conservation of momentum for a fluid medium. In a general form, the Navier-Stokes equations are given as follows:

1. Continuity Equation (Conservation of Mass)

$$\partial\rho/\partial t + \nabla\cdot(\rho u) = 0$$

where ρ represents the fluid density, t represents time, u represents the velocity vector, $\partial/\partial t$ represents the partial derivative with respect to time and ∇ represents the divergence operator.

2. Momentum Equation (Conservation of Momentum)

$$\rho(\partial u/\partial t + u\cdot\nabla u) = -\nabla p + \mu\nabla^2 u + \rho g$$

where p represents pressure, μ represents dynamic viscosity and g represents the acceleration due to gravity.

These equations are often accompanied by additional equations, depending on the specific modelling assumptions and physical phenomena being considered[2]. For instance, incompressible flow assumes that density remains constant, simplifying the continuity equation to $\nabla\cdot u = 0$. Turbulent flows may require the inclusion of turbulence models such as the Reynolds-averaged Navier-Stokes (RANS) equations or Large Eddy Simulation (LES) equations.

In summary, the governing equations of CFD are essential tools for simulating fluid flows. These equations are the foundations upon which CFD simulations are based, allowing engineers and scientists to analyse and predict fluid behaviour in various scenarios[24].

Analysis through Ansys integrating CFD

Creating a geometry in design modeler and domain is discretized with effective mesh is a critical aspect of conducting Computational Fluid Dynamics (CFD) analysis, particularly in the context of dam break simulations[23]. The importance of mesh creation is closely tied to ensuring accurate and reliable results and it involves considerations of boundary conditions and flow initialization[7,18,32]. The given details highlight various aspects of mesh creation and its significance:

1. Mesh Quality and Resolution: Mesh generation begins with defining the number of nodes and elements. In the case presented, the initial mesh consists of 42,282 nodes and 41,680 elements presented in figure 3. The choice of mesh resolution is pivotal; a coarse mesh might lead to inexact results while a fine mesh can be computationally expensive[25]. Striking the right balance is crucial to obtain meaningful results.

2. Accuracy of Boundary Conditions: Accurate boundary conditions, as depicted in figure 4, are fundamental for realistic simulations[22,39]. In the dam break analysis, it is essential to specify the boundary conditions correctly to mimic the physical behavior of the fluid near the dam including inlet velocities and pressure outlets. Incorrect boundary conditions can significantly impact the simulation results.

3. Flow Initialization: Flow initialization, as indicated in figure 5, involves setting the initial conditions for the simulation. In this case, a flow of 25 meters is considered on a 1:10 scale. Proper flow initialization ensures that the simulation starts from a state representative of the real-world scenario[8,33,42]. It prevents the simulation from starting with unrealistic or arbitrary conditions that can lead to unphysical results.

4. Temporal Analysis: The provided information mentions that the CFD analysis covers various time steps from 1s to 5s. The mesh should be able to capture the evolving flow dynamics over these time intervals accurately. Mesh adaptability or refinement may be necessary to account for changes in flow characteristics as the dam break scenario progresses[19,37,38]. In summary, mesh creation, boundary condition definition and flow initialization are integral steps in conducting CFD-based dam break analysis. The quality and accuracy of these components directly influence the credibility and usefulness of the simulation results which, in turn, can inform decision-making processes and improve our understanding of complex fluid dynamics scenarios.

Results and Discussion

In the realm of dam break analysis, obtaining a profound understanding of fluid dynamics is paramount for ensuring safety, optimizing infrastructure design and managing potential risks. To this end, we delve into a series of insightful figures that shed light on the intricate details of a dam break scenario. These figures, captured at a critical juncture - 1 meter from the dam site with a flowing velocity of 1 m/s, offer a glimpse into the evolving fluid dynamics. They enable us to grasp the magnitude, direction and distribution of velocities, the pressures exerted on surfaces, the kinetic energy of turbulence and the overall energy state within the fluid[1,9,20]. Each figure unveils a unique facet of the complex interplay between water, structures and the environment during a dam break event, empowering engineers, researchers and decision-makers with invaluable data to guide their assessments and design considerations[17].

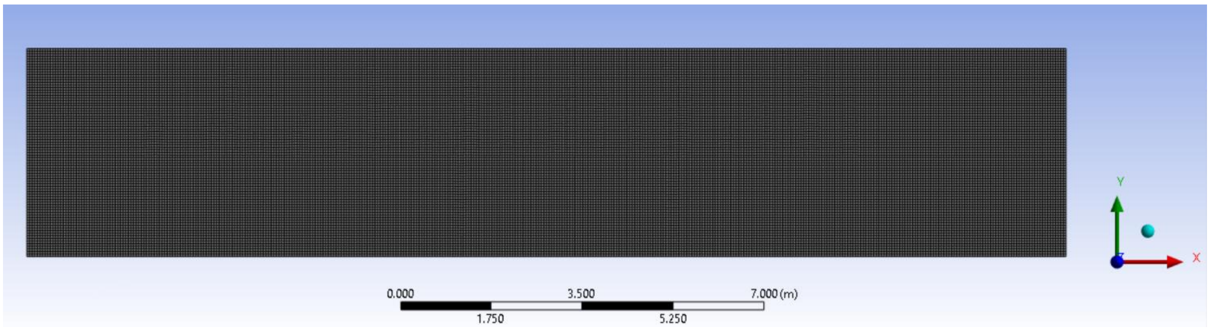


Figure 3: Initial mesh configuration

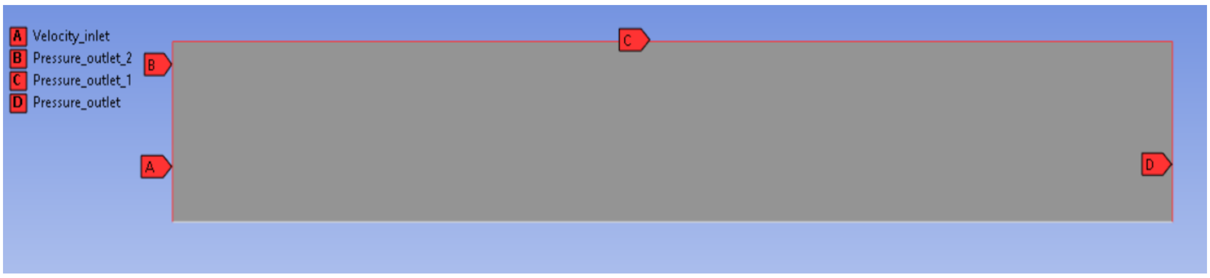


Figure 4: Boundary condition setup

https://doi.org/10.25303/173da035047

Figure 5: Flow initialization

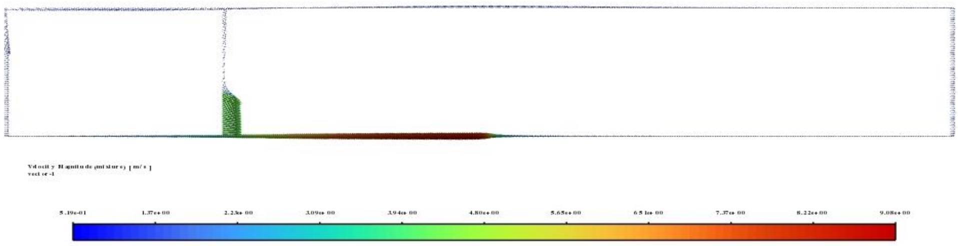


Figure 6: Vectors of velocity magnitude at 1 second

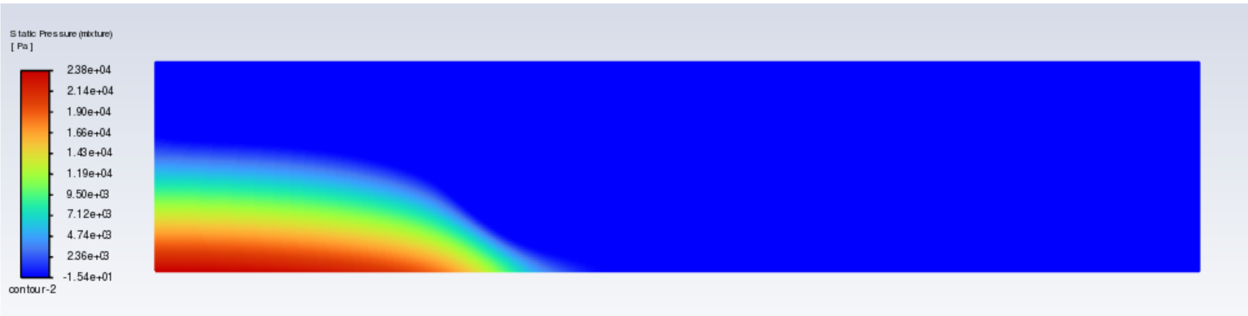


Figure 7: Static pressure at 1 second

Several key parameters namely Static Pressure, Dynamic Pressure, Total Pressure, Velocity Magnitude and Flow Depth at intervals of 1, 2, 5 and 7 meters (scaled at 1:10) have been assessed from the dam site for 1s, 2s, 2.5s, 3s and 5s respectively[21]. Figure 6 showcases the vectors representing the magnitude and direction of fluid velocities at a critical moment, 1 second after the dam break[15,45]. This visualization allows us to grasp how the fluid is moving, its speed and the formation of flow patterns. It provides insights into the early dynamics of the dam break scenario.

Static Pressure: Figure 7 presents the static pressure distribution. This information is vital for understanding the pressure forces acting on structures or surfaces within the flow field. It aids in assessing the structural integrity of dams or other engineering structures and helps in designing them to withstand the pressure loads created by the flowing fluid.

Dynamic Pressure: Figure 8 focuses on dynamic pressure which accounts for the kinetic energy of the fluid. It reveals regions of high and low dynamic pressure within the flow field. This data is crucial for evaluating the impact forces that the fluid exerts on objects in its path and is essential for dam safety assessments and hydraulic engineering.

Figure 9 illustrates the total pressure distribution. Total pressure combines static and dynamic pressure and is important for assessing the overall energy within the fluid[14]. Monitoring total pressure helps in understanding the energy changes as the fluid flows and can inform design choices for hydraulic systems and infrastructure.

Figure 10 provides insights into the turbulence kinetic energy present in the flow field. Turbulence can significantly affect the stability of structures and the mixing of substances within the fluid. This figure aids in evaluating the turbulence intensity and identifying zones where turbulence may be a concern[16]. These figures collectively offer a comprehensive view of the fluid dynamics resulting from the dam break, providing critical data for engineering, safety and environmental assessments[26]. They help engineers and researchers make informed decisions and develop mitigation strategies to manage the consequences of such events effectively.

https://doi.org/10.25303/173da035047

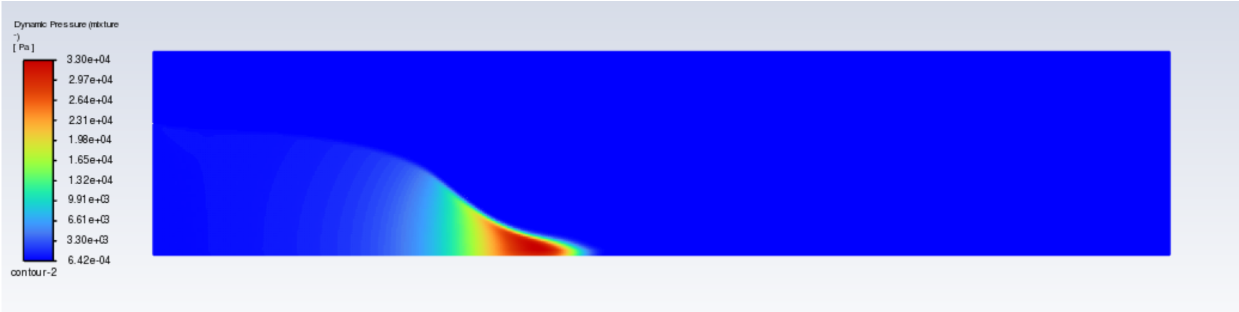


Figure 8: Dynamic pressure at 1 second

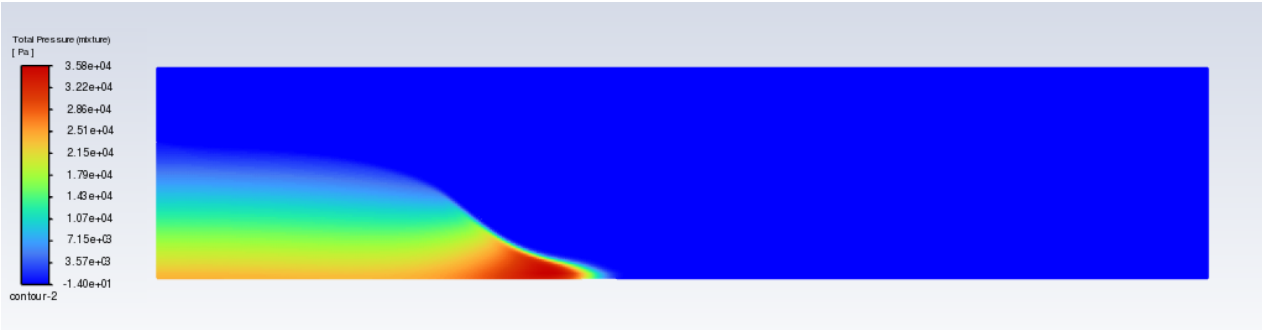


Figure 9: Total pressure at 1 second

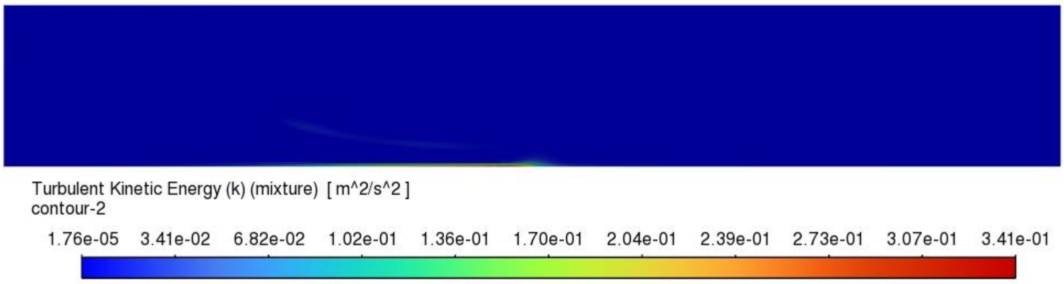


Figure 10: Turbulence kinetic energy at 1 second

Table 1
Evaluation of flow parameters at different spacings (1s, 2s, 2.5s, 3s, 5s) at 1-meter distance from dam break site

S. N.	Parameters	1s	2s	2.5s	3s	5s
1	Static pressure (pa)	6924.84	6390.32	6142.49	5918.31	5563.13
2	Dynamic pressure (pa)	8839.52	6624.85	6099.40	5674.86	5179.26
3	Total pressure (pa)	15764.36	13015.17	12241.89	11593.17	10742.39
4	Velocity Magnitude (m/s)	4.23	3.65	3.51	3.38	3.18
5	Depth of the flow (m)	2.10	2.30	2.40	2.50	2.55

Table 2
Evaluation of flow parameters at different spacings (2s, 2.5s, 3s, 5s) at 2-meter distance from dam break site

S. N.	Parameters	2s	2.5s	3s	5s
1	Static pressure (pa)	5541.23	5543.30	5457.36	5050.46
2	Dynamic pressure (pa)	8826.21	7656.53	6851.07	6039.09
3	Total pressure (pa)	14367.44	13199.83	12308.43	11089.55
4	Velocity Magnitude (m/s)	4.22	3.93	3.71	3.46
5	Depth of the flow (m)	1.98	1.78	1.64	1.53

In a comprehensive analysis of flow parameters following a dam break incident, various spacings and distances from the dam break site were considered. Table 1 reveals the evaluation of flow parameters at distances of 1 meter from the site, with spacings ranging from 1 second to 5 seconds. Static pressure exhibited a decreasing trend with increasing spacing, starting at 6924.84 Pa at 1s and decreasing to 5563.13 Pa at 5s. Similarly, dynamic pressure, total pressure, velocity magnitude and flow depth showed a corresponding trend. Table 2 extends the analysis to a 2-meter distance, focusing on spacings of 2s to 5s, demonstrating similar decreasing trends in flow parameters[27].

Table 3 explores parameters at 5 meters from the dam break site and table 4 at 7 meters, with both tables showing the same trend. These findings are crucial for dam break risk assessment, providing insights into how flow parameters evolve with distance and time since the dam breach. Further analysis and modeling based on these data will be essential for improving dam safety measures and emergency response planning[28].

Significance of contour analysis in dam break assessment: Drawing contours including contour pressure and contour velocity, at different spacing intervals is essential in dam break analysis for several crucial reasons[31]. Contour drawings provide a visual representation of the distribution of parameters such as pressure and velocity across a given area affected by a dam break. By displaying this information graphically, engineers and researchers can gain immediate insights into the spatial variations of these critical variables. This visual representation helps in identifying regions of high or low pressure and velocity, enabling a more comprehensive understanding of the flow dynamics and potential hazards. Secondly, contour pressure maps are invaluable for assessing the safety and stability of structures and communities near the dam site.

By analyzing these contours, experts can identify areas that are subject to high pressure, indicating potential risks of structural failure or flooding. This information is vital for emergency response planning, as it allows for the prioritization of evacuation efforts and the deployment of resources to mitigate the impact of the dam break.

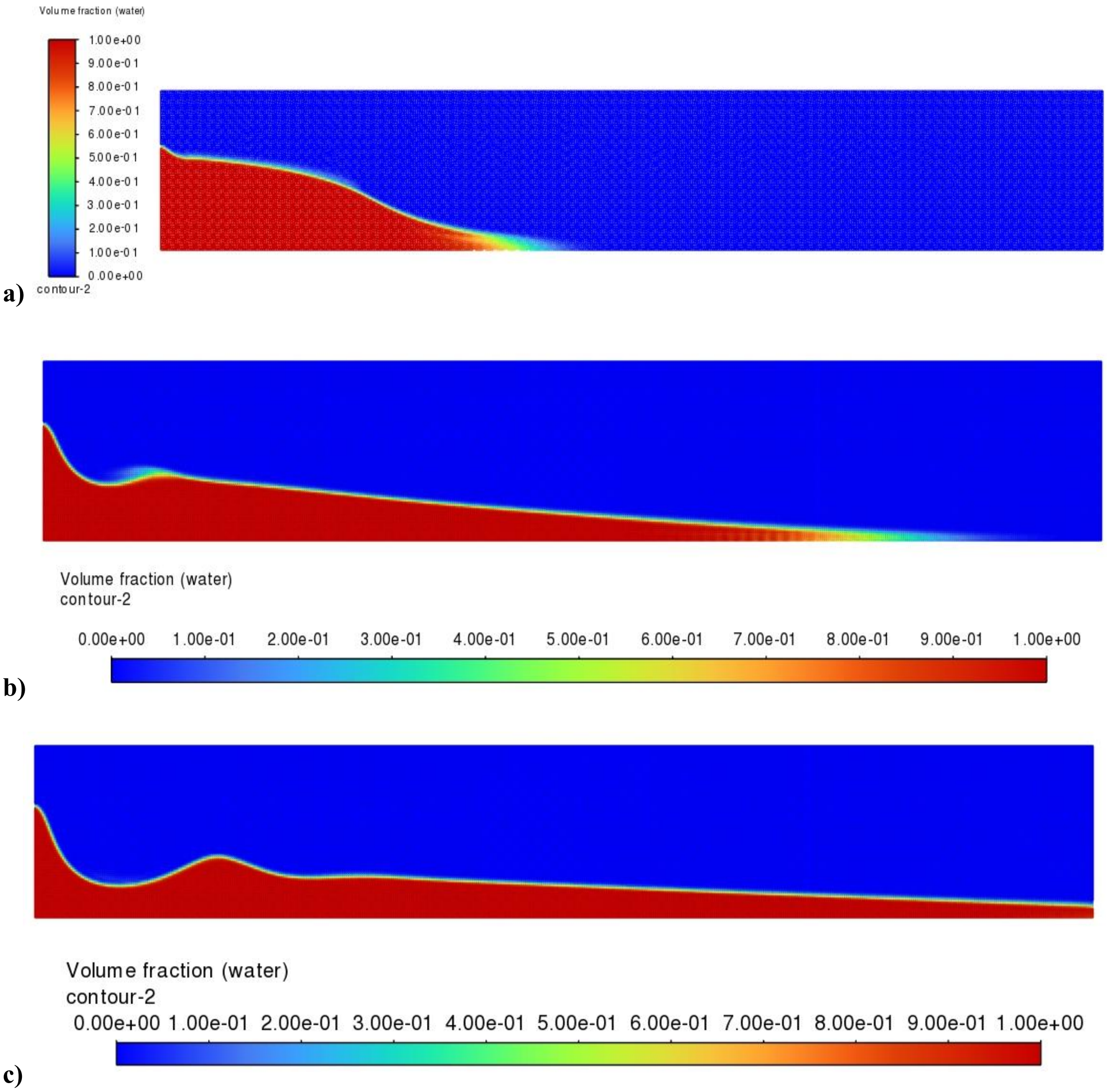


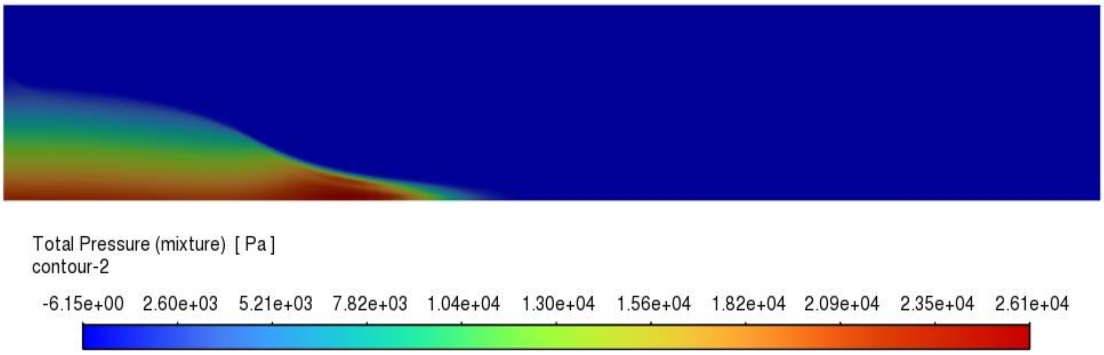
Figure 11: Contours at a) 1s, b) 3s and c) 5s respectively.

Table 3
Evaluation of flow parameters at different spacings (2.5s, 3s, 5s) at 5-meter distance from dam break site

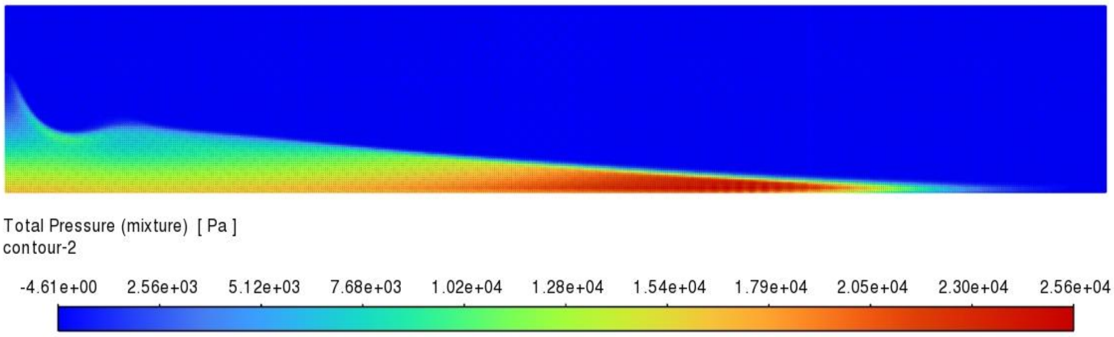
S. N.	Parameters	2.5s	3s	5s
1	Static pressure (pa)	2906.91	3356.67	3995.56
2	Dynamic pressure (pa)	16985.92	14274.08	8746.59
3	Total pressure (pa)	19892.83	17630.75	12742.15
4	Velocity Magnitude (m/s)	5.89	5.37	4.20
5	Depth of the flow (m)	1.63	1.56	1.43

Table 4
Evaluation of flow parameters at different spacings (3s and 5s) at 7-meter distance from dam break site

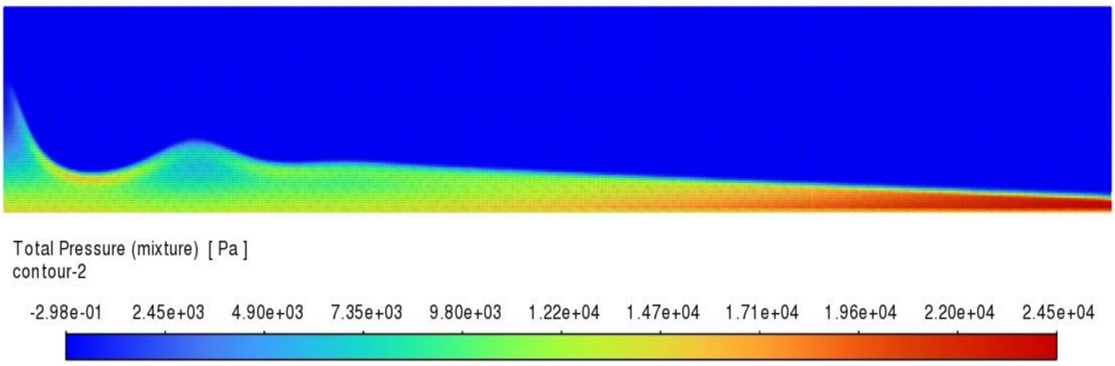
S. N.	Parameters	3s	5s
1	Static pressure (pa)	1097.09	2672.99
2	Dynamic pressure (pa)	18776.83	14443.31
3	Total pressure (pa)	19873.92	17116.30
4	Velocity Magnitude (m/s)	7.25	5.41
5	Depth of the flow (m)	1.58	1.42



a)

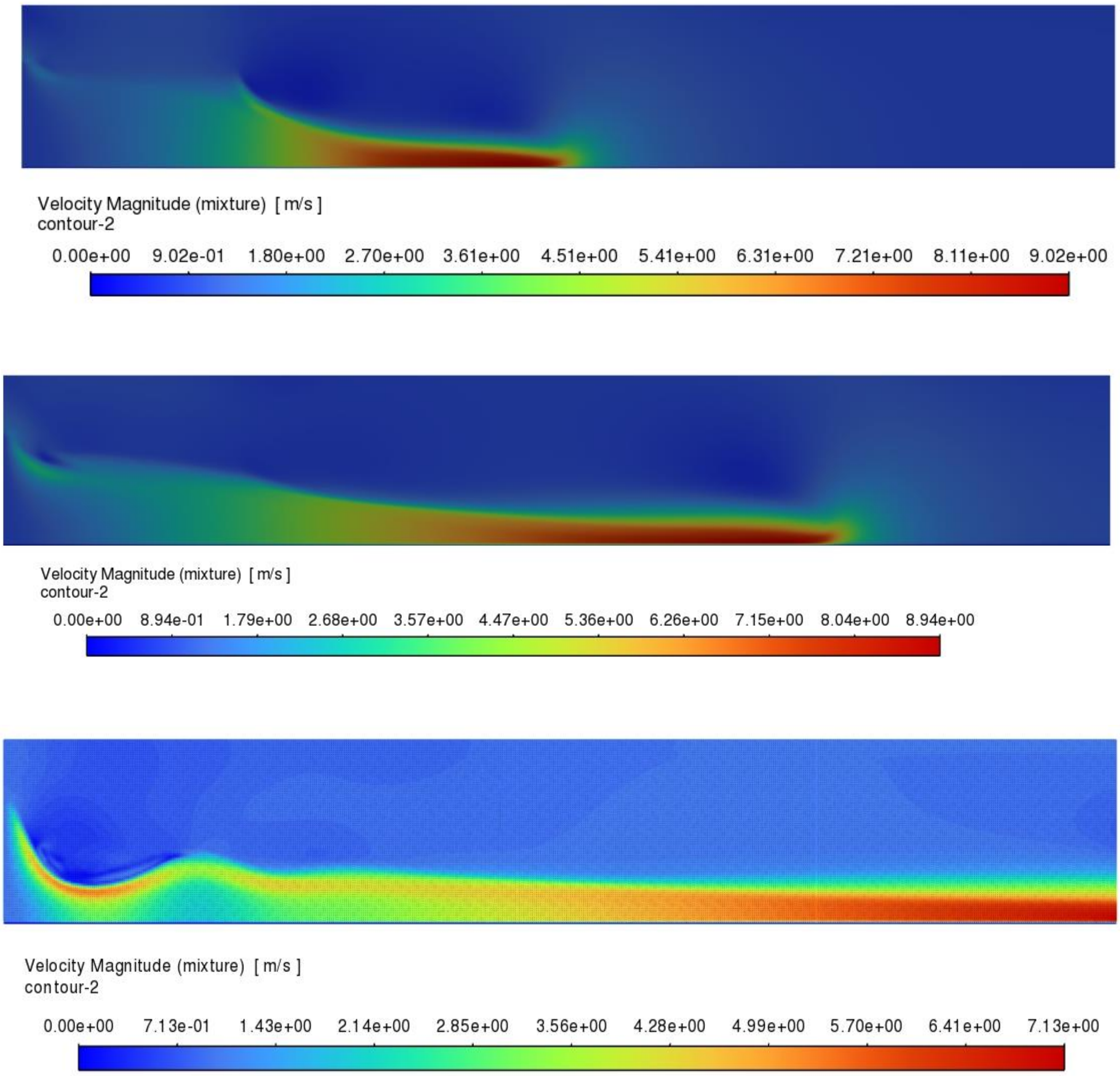


b)



c)

Figure 12: Contour pressure at a) 1s, b) 3s and c) 5s respectively.



a)

b)

c)

Figure 13: Contour velocities at a) 1s, b) 3s and c) 5s respectively.

Contour velocity maps are equally important because they reveal how fast the water is moving across the affected area[34]. Understanding the velocity distribution is crucial for predicting the speed at which floodwaters will reach different locations downstream. This information can help for evacuation orders and help authorities allocate resources for rescue and relief operations effectively.

Furthermore, varying the spacing between contour lines or grids allows for a more detailed analysis of the dam break scenario. Narrower spacing provides a higher level of detail which is particularly useful for identifying small-scale variations in pressure and velocity. Wider spacing, on the other hand, offers a broader overview of the flow patterns, making it easier to identify large-scale trends and hazards[41]. Having the ability to adjust the spacing of contours provides flexibility in the analysis, ensuring that both macro and micro-level features are considered.

In conclusion, drawing contours, contour pressure and contour velocity for different spacing intervals in dam break analysis are crucial for assessing the distribution of pressure and velocity, identifying potential hazards and informing emergency response efforts. These visual representations are essential tools for engineers, researchers and emergency responders, enabling them to make informed decisions to safeguard lives and property in the event of a dam break. The contour images, depicting contour pressure and contour velocity, have been generated at time intervals of 1s, 3s and 5s as illustrated in figures 11, 12 and 13 respectively.

Conclusion

Based on the data and analysis presented in this study, several important conclusions can be drawn regarding dam break scenarios and their potential impact on flow parameters at different distances from the dam site.

1. Temporal evolution of flow parameters: The data collected at various time intervals (1s, 2s, 2.5s, 3s and 5s) at a 1-meter distance from the dam break site reveals the dynamic nature of flow parameters. Notably, there is a consistent decrease in static pressure and dynamic pressure over time, reflecting the dissipation of pressure as the water flows away from the dam. Additionally, the velocity magnitude decreases with time, indicating a reduction in the flow velocity. This temporal evolution is critical for understanding the progression of a dam break event.

2. Spatial variations in flow parameters: The study extends its analysis to different distances from the dam break site (1m, 2m, 5m and 7m). One significant observation is that static pressure decreases as the distance from the dam site increases, while dynamic pressure shows a similar trend. This spatial variation highlights the importance of considering the impact of dam break events on structures and areas situated at different distances from the dam.

3. Total pressure and its implications: The total pressure, which accounts for both static and dynamic pressure, follows a similar trend, decreasing with time and distance from the dam site. This parameter is crucial for assessing the overall energy in the flow. The decline in total pressure underscores the diminishing force of the water as it moves away from the dam, indicating a lower potential for damage and destruction further downstream.

4. Velocity magnitude and flow depth: The velocity magnitude data at various distances from the dam site provide insights into the speed at which floodwaters travel. This information is vital for estimating the arrival time of the flood at different locations and aids in emergency response planning. Additionally, the data on flow depth at different distances reveal how the depth of the flow changes with time and distance, further aiding in understanding the flow dynamics.

5. Importance of contour analysis: The use of contour analysis is pivotal in visualizing and interpreting the spatial distribution of flow parameters. It allows for a clear depiction of how pressure and velocity vary across the affected area, enabling precise identification of high-pressure zones and areas with potentially hazardous flow velocities. This visual representation is invaluable for emergency response planning and risk assessment.

In conclusion, this study's comprehensive analysis of dam break scenarios using CFD and GIS offers valuable insights into the dynamic nature of flow parameters over time and distance. These insights are essential for enhancing our understanding of dam break events, improving risk assessment and mitigation strategies and ultimately contributing to the safety of communities and infrastructure near dams.

References

1. Ahmadi A. and Ghodsian M., Numerical modeling of dam-break flows with sediment transport using a modified VOF method, *Journal of Hydraulic Research*, **53(5)**, 656-669 **(2015)**

2. Ashgriz N. and Mostaghimi J., An introduction to computational fluid dynamics, Fluid Flow Handbook, **1**, 1-49 **(2002)**

3. Bellos C.V., Soulis J.V. and Sakkas J.G., Computation of two-dimensional dam-break-induced flows, *Advances in Water Resources*, **14(1)**, 31-41 **(1991)**

4. Bhatt C.M., Srinivasa Rao G., Manjushree P. and Bhanumurthy V., Space based disaster management of 2008 Kosi floods, North Bihar, India, *Journal of the Indian Society of Remote Sensing*, **38**, 99-108 **(2010)**

5. Bhukya M.N. and Kota V.R., A quick and effective MPPT scheme for solar power generation during dynamic weather and partial shaded conditions, *Engineering Science and Technology, an International Journal*, **22(3)**, 869-884 **(2019)**

6. Biscarini C., Di Francesco S. and Manciola P., CFD modelling approach for dam break flow studies, *Hydrology and Earth System Sciences*, **14(4)**, 705-718 **(2010)**

7. Chang T.J., Kao H.M., Chang K.H. and Hsu M.H., Numerical simulation of shallow-water dam break flows in open channels using smoothed particle hydrodynamics, *Journal of Hydrology*, **408(1-2)**, 78-90 **(2011)**

8. Fang M. and Li W., Large eddy simulation of dam-break flow with SPH method, *Environmental Fluid Mechanics*, **19(4)**, 959-978 **(2019)**

9. Feng Y. and Xie J., Numerical simulation of dam break flows over an erodible bed using a two-phase SPH model, *Environmental Fluid Mechanics*, **18(2)**, 365-388 **(2018)**

10. Gade M.S.L. and Rooban S., Run time fault tolerant mechanism for transient and hardware faults in alu for highly reliable embedded processor, International Conference on Smart Technologies in Computing, Electrical and Electronics (ICSTCEE), 44-49 **(2020)**

11. Kalyani G., Janakiramaiah B., Prasad L.N., Karuna A. and Babu A.M., Efficient crowd counting model using feature pyramid network and ResNeXt, *Soft Computing*, **25(15)**, 10497-10507 **(2021)**

12. Kocaman Selahattin and Kaan Dal, A new experimental study and SPH comparison for the sequential dam-break problem, *Journal of Marine Science and Engineering*, **8(11)**, 905 **(2020)**

13. Kumar S., Jaswal A., Pandey A. and Sharma N., Literature review of dam break studies and inundation mapping using hydraulic models and GIS, *International Research Journal of Engineering and Technology*, **4(5)**, 55-61 **(2017)**

14. Kumari C.U., Murthy A.S.D., Prasanna B.L., Reddy M.P.P. and Panigrahy A.K., An automated detection of heart arrhythmias using machine learning technique: SVM, Materials Today: Proceedings, **45(1)**, 1393-1398 **(2021)**

15. Li J. and Yu Y., Numerical simulation of dam-break flow over an erodible bed using a two-phase SPH model, *Environmental Fluid Mechanics*, **15(5)**, 959-977 **(2015)**

16. Li X. and Zhao J., Dam-break of mixtures consisting of non-Newtonian liquids and granular particles, *Powder Technology*, **33(8)**, 493-505 **(2018)**

17. Li Z. and Wang Z., Numerical simulation of dam-break flow over movable beds using a finite element model, *Advances in Water Resources*, **94**, 233-244 **(2016)**

18. Lin B. and Chen H., Application of CFD in dam-break flood simulation and risk assessment, *Journal of Hydrodynamics*, **29(2)**, 249-259 **(2017)**

19. Lin P. and Lin B., Numerical simulation of dam-break flow with sediment transport using Smoothed Particle Hydrodynamics, *Journal of Hydrodynamics*, **27(2)**, 201-214 **(2015)**

20. Liu G. and Yang S., Numerical modeling of dam-break flow with sediment transport using a hybrid finite volume/SPH method, *Advances in Water Resources*, **70**, 31-44 **(2014)**

21. Lu Y. and Borthwick A.G., Numerical modeling of dam-break flows over mobile beds with entrainment, *Journal of Hydraulic Engineering*, **143(5)**, 04017008 **(2017)**

22. Ma Y., Zhang H., Yu Z. and Li Y., Numerical simulation of dam-break flows with complex geometry using an improved SPH method, *Journal of Hydrodynamics*, **27(6)**, 933-944 **(2015)**

23. Matsson J.E., An Introduction to Ansys Fluent 2023, Sdc Publications **(2023)**

24. McLay A., Computational Fluid Dynamics: the Basics with Applications, JD Anderson, McGraw-Hill Book Company Europe, McGraw-Hill House, *The Aeronautical Journal*, **100(998)**, 365-365 **(1996)**

25. Moulinec C. and Escudier M., An artificial boundary condition for direct numerical simulation of compressible viscous flows in complex geometries, *Journal of Computational Physics*, **162(2)**, 338-371 **(2000)**

26. Nagarjuna T., Lakshmi S. and Nehru K., USRP 2901-based SISO-GFDM transceiver design experiment in virtual and remote laboratory, *The International Journal of Electrical Engineering and Education*, **59(4)**, 350-365 **(2022)**

27. Padmaja B., Myneni M.B. and Krishna Rao Patro E., A comparison on visual prediction models for MAMO (multi activity-multi object) recognition using deep learning, *Journal of Big Data*, **7(1)**, 1-15 **(2020)**

28. Pon V.D., Wilson K.J., Hariprasad K., Ganesh V., Ali H.E., Algarni H. and Yahia I.S., Enhancement of optoelectronic properties of ZnO thin films by Al doping for photodetector applications, *Superlattices and Microstructures*, **151(1)**, 106790 **(2021)**

29. Qin Y. and Chen Q., Numerical simulation of dam-break flow over an erodible bed with a modified two-layer model, *Journal of Hydrodynamics*, **31(6)**, 1116-1127 **(2019)**

30. Raji A. et al., Biowaste-originated heteroatom-doped porous carbonaceous material for electrochemical energy storage application, *Journal of Industrial and Engineering Chemistry*, **98(1)**, 308-317 **(2021)**

31. Ramya N.S. and Kumar N.S., Analysis and Simulation of Flow over Stepped Spillway Using Ansys-CFD, *Asian Research Journal of Current Science*, **5(1)**, 155-162 **(2023)**

32. Saravanan P., Prethivirajan A.R., Sivaprasanna A.S., Udhayakumar K. and Sivapragasam C., Performance Evaluation of an ANN based Bias Correction algorithm in Monthly and Daily Precipitation Time Series of La Farge Station, USA, *Disaster Advances*, **16(4)**, 27-33 **(2023)**

33. Soares-Frazao S., Canelas R., Cao Z., Cea L., Chaudhry H.M., Die Moran A., El Kadi K., Ferreira R., Cadórniga I.F., Gonzalez-Ramirez N. and Greco M., Dam-break flows over mobile beds: experiments and benchmark tests for numerical models, *Journal of Hydraulic Research*, **50(4)**, 364-375 **(2012)**

34. Spandana K. and Rao V.S., Internet of things (Iot) based smart water quality monitoring system, *International Journal of Engineering & Technology*, **7**, 259-262 **(2018)**

35. Stern F., Wilson R.V., Coleman H.W. and Paterson E.G., Comprehensive approach to verification and validation of CFD simulations—part 1: methodology and procedures, *J. Fluids Eng.*, **123(4)**, 793-802 **(2001)**

36. Sun R., Wang X., Zhou Z., Ao X., Sun X. and Song M., Study of the comprehensive risk analysis of dam-break flooding based on the numerical simulation of flood routing, Part I: model development, *Natural Hazards*, **73(3)**, 1547–1568 **(2014)**

37. Vignaga E., Gallati M. and De Cesare G., Experimental and numerical investigation of dam-break flow over erodible beds, *Journal of Hydraulic Research*, **57(1)**, 89-106 **(2019)**

38. Wang Y. and Huang W., Numerical simulation of dam-break flow over an erodible bed using a modified SPH method, *Journal of Hydrodynamics*, **29(5)**, 727-739 **(2017)**

39. Xie S. and Liang D., Numerical simulation of dam-break flows on a movable bed with sediment transport using SPH method, *International Journal of Computational Fluid Dynamics*, **32(6)**, 267-288 **(2018)**

40. Xue Z., You J. and Wang Z., Dam-break flows with sediment transport: a review, *Journal of Hydrodynamics*, **26(2)**, 175-188 **(2014)**

41. Yue L., Jayapal M., Cheng X., Zhang T., Chen J., Ma X., Dai X., Lu H., Guan R. and Zhang W., Highly dispersed ultra-small nano Sn-SnSb nanoparticles anchored on N-doped graphene sheets as high performance anode for sodium ion batteries, *Applied Surface Science*, **512(1)**, 145686 **(2020)**

42. Zainol Abidin M.S. and Etemad-Shahidi A., Application of finite volume method for dam-break flow over mobile bed, *Journal of Hydraulic Research*, **51(6)**, 675-686 **(2013)**

43. Zhang H. and Li Z., Application of CFD in dam break flood simulation and risk assessment, *Journal of Hydroinformatics*, **19(2)**, 309-324 **(2017)**

44. Zhang L. and Song C., Numerical simulation of dam-break flows over mobile bed, *Advances in Water Resources*, **95(1)**, 1-15 **(2016)**

45. Zhang Z. and Zong Z., Numerical modeling of dam-break flow over erodible beds with a VOF-DEM coupled method, *Computers & Fluids*, **170(1)**, 233-248 **(2018)**

46. Zhou Zhengyin, Xiaoling Wang, Ruirui Sun, Xuefei Ao, Xiaopei Sun and Mingrui Song, Study of the comprehensive risk analysis of dam-break flooding based on the numerical simulation of flood routing. Part II: Model application and results, *Nat Hazards*, **72(1)**, 675–700 **(2014)**.

(Received 04th October 2023, accepted 05th December 2023)